eta/VPG
TRAINING MANUAL

A LS/DYNA based Full Vehicle Simulation Solution Package.

Version: 1.0
Release Date: August 19, 1998

FORWARD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Engineering Technology Associates, Inc.
Creation Date: January 15, 1997
Revision Level: 1
Revision Date: April 7, 1997
Approved by: Bob Ondrus

eta/VPG
Training Manual

Version 1.0

Engineering Technology Associates, Inc.
1100 East Mandoline, Suite B
Madison Heights, MI 48071

Phone: (248) 588-9010
Support: (800) ETA-FEMB
Fax: (248) 588-9014

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/VPG are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of the respective owners.

Copyright 1998, 1999 Engineering Technology Associates, Inc. All rights reserved.

TABLE OF CONTENTS

Introduction 1

eta/VPG Menu System 1

Opening/Creating an eta/VPG Database File 5

Pre-Processing

1- Auto-Generating a **Front Suspension** Model 7

2- Auto-Generating a **Rear Suspension** Model 11

3- Auto-Generating **Tire** Models 15

- Attaching Tires to Suspension 18
- Defining Airbags for Tire Pressure 19
- Defining Initial Rotational Velocity 20

4- Auto-Generating a **Road** Surface 22

- Defining Contact between Road and Tires 25
- Constraining the Road 26
- Assigning Velocity to the Road 26

5- Attaching a **Body Model** to Chassis/Suspension model 28

- Defining Gravity 30
- Pre-loading the Suspension 31

Dynamic Analysis

6- Defining VPG Analysis Control Parameters 33

7- Analysis Submission 38

Post-Processing

8- Displaying Results 40

9- Graph Plotting 46

Conclusion

49

Introduction

This training manual was created to familiarize new users with the eta/Virtual Proving Ground(VPG) software. The exercise will show the user how to Auto-Generate suspension(front/rear), tire, and road models using the VPG libraries. A Simplified Body will be added, the full system model will be analyzed and the results post-processed.

eta/VPG Menu System

The menu system is an open system. Most operations are accessible at any time during the eta/VPG session. The user interface consists of four parts: the Drawing window, Menu window, Dialogue window, View Options window, and Display Parameter Options window.

The menu system is set up as a tree structure. The program starts in the MAIN menu and branches out into sub-menus. The user selects a sub-menu by mouse pick or keyboard entry. Descriptions for these menu options are located in their respective sections.

FILE	Importing and exporting data files. (i.e. LS/DYNA, NASTRAN input files.)
PRE-PROCESSOR	This menu includes the following submenus: LINE - generation and manipulation of geometry(lines/points). SURFACE - generation and manipulation of surfaces. ELEMENT - generation and manipulation of finite elements. NODE - generation and manipulation of nodes.

MODEL CHECKER
- commands to check a models' integrity.

SET MENU
- commands to assign nodes and elements to sets.

BOUNDARY CONDITION
- commands to define loads, initial velocities, spc's, etc.

CONSTRAINT
- commands to define LS/DYNA contraint cards.

MATERIAL
- commands to define LS/DYNA material property cards.

ELEMENT PROPERTY
- commands to define LS/DYNA element property cards.

CONTACT INTERFACE
- commands to define LS/DYNA Contact Interface cards.

DYNA MISC.
- commands to define LS/DYNA control cards.

ROAD	This menu contains a library of road surface data from which the user can select, or define their own.
SUSPENSION	This menu contains a library of front and rear suspensions from which the user can select, or define their own.
TIRE MODEL	This menu allows the user to Auto-Generate tire models based on user defined tire geometry parameters.
ANALYSIS	The user can define LS/DYNA analysis control parameters, then submit the job to run in the background interactively, or write an input file directly.

FATIGUE	
FFT	
POST PROCESSING	Menu that contains commands for viewing analysis results(deformed shape/contour plotting & animation).
GRAPH	LS/DYNA ASCII result files, which contain the dynamic characteristics of the analyzed model vs. time, can be read and plotted. Once read, the data can be manipulated using an array of commands.
RESTART	Allows the user to restart (**without saving the current database**) from the current, a new, or an existing database.
SESSION FILE	Allows the user to regain lost work (i.e., system crash) thru reading a short executable that is saved in a file called VPG.ses.
VIEW	Commands for manipulating a model's display, position, and perspective in eta/VPG.
EXIT PROGRAM	Exits eta/VPG.

Function keys 1 thru 8 are also available to jump to the most frequently used menus.
The F1 (function key 1) is reserved for the MAIN menu.

F1	Main Menu	F5	Model Checker
F2	Element Options	F6	Node Options
F3	File Manager	F7	Surface Options
F4	Geometry Builder	F8	Free Rotation

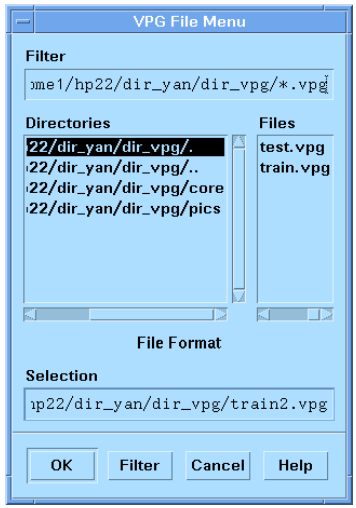
eta/VPG is composed of 5 motif widows. The 5 windows are described and illustrated below. the Drawing window, Menu window, Dialogue window, View Options window, and Display Parameter Options window.

1. Drawing Window - geometry/models and graphs are displayed in this window.
2. Menu Window - menus are listed. Menus can be selected with a left mouse click.
3. Dialogue Window - messages are displayed here, and commands can be typed at the prompt.
4. View Options window - the PART, SETUP, and UTILITIES menus are selectable here, along with view manipulation commands.
5. Display Parameters Options window - display parameters such as shading, fill color/hide, along with REDRAW and RESET are selectable.

Opening/Creating an eta/VPG Database File

Type **'vpg'** in a Unix window to start the program. Once the windows are activated, the VPG FILE MENU window is displayed for the user to OPEN or CREATE a new VPG database.

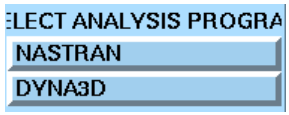
1.



2. The user would either select the name of a previously saved file or enter the name of a new file in the Dialogue window. The recommended practice is to add the extension **.vpg** to a newly created file.

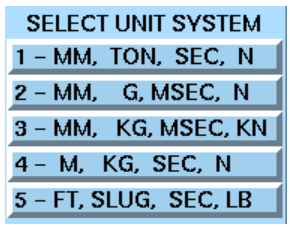
3. If creating a NEW file, the user would be prompted to do so:

4. The user will be prompted to select the analysis program desired:



The analysis program selected will set defaults for the eta/VPG session to generate either LS/DYNA or NASTRAN cards.

5. The user will also be prompted to select the Unit System desired:

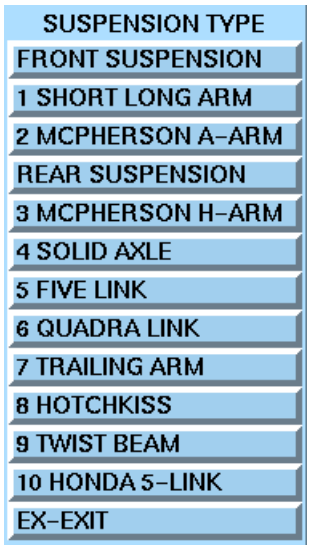


The user will now be in the MAIN MENU of eta/VPG and ready to start the session.

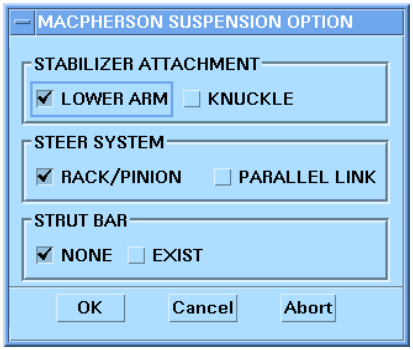
1- Auto-Generating a Front Suspension Model

1- From the Main menu, select the SUSPENSION menu.

2- Select CREATE. A suspension menu will be displayed from which the user can select different types of front and rear suspension models.

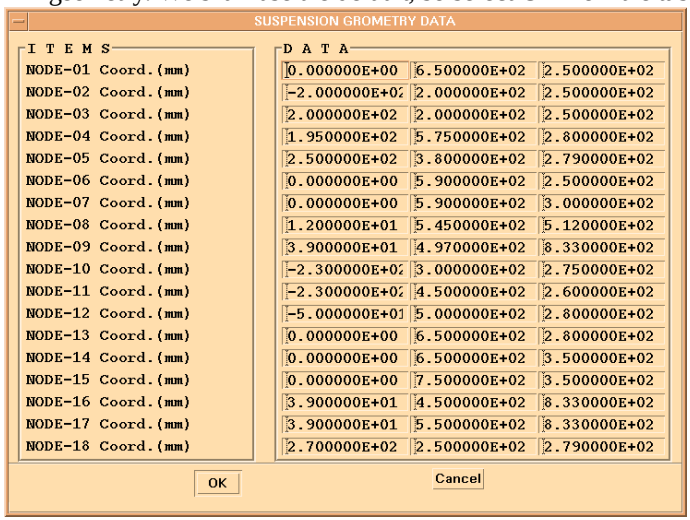


3- For the FRONT SUSPENSION, select type:
2- McPherson A-ARM



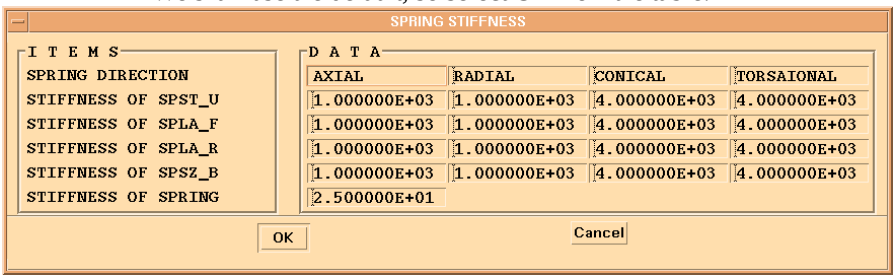
The user can select the Steering System, Stabilizer types and whether or not there is a Strut Bar in the above table.

4- The suspension geometry will be displayed along with a table listing the coordinates of nodes that define the geometry. The user can edit this table to define their particular suspension geometry. We shall use the default, so select OK from the table.

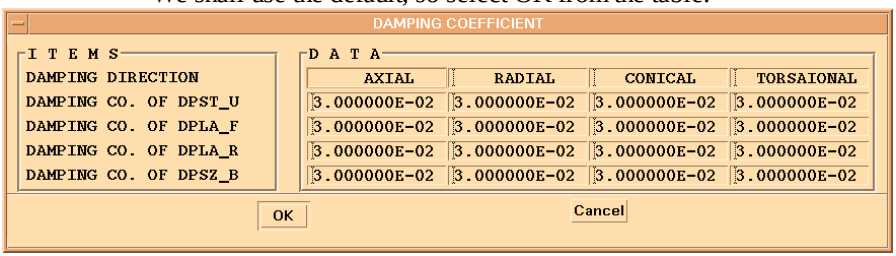


a- eta/VPG will prompt:
'ACCEPT THESE GEOMETRY DATA?(Y/N)'
b- answer Yes.

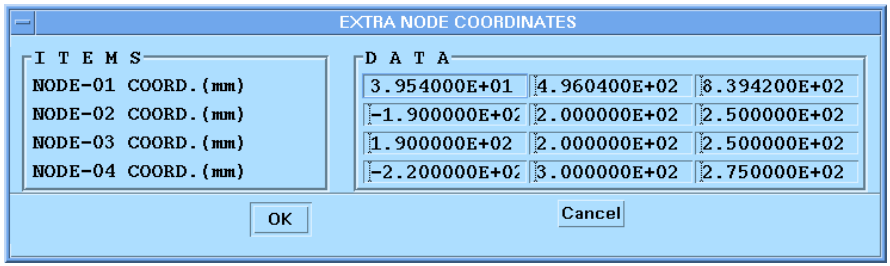
5- A Spring Stiffness menu will be displayed in which the user can specify the spring rates of the suspension. Again, the user can edit this table to define their particular spring rates.
We shall use the default, so select OK from the table.



6- A Damping Coefficient menu will then be displayed in which the user can specify the bushing rates of the suspension. Again, the user can edit this table to define their particular bushing rates.
We shall use the default, so select OK from the table.

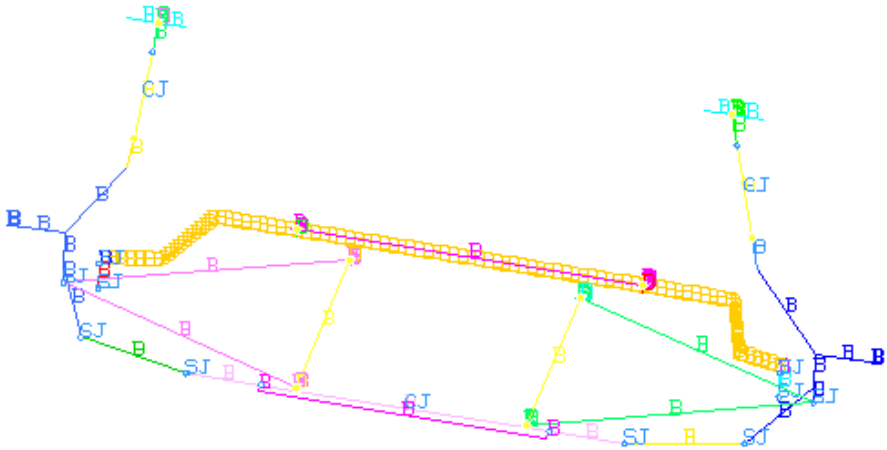


7- An Extra Node Coordinate menu will then be displayed in which the user can specify the coordinates of the body attachment points of the suspension. Again, the user can edit this table to define their particular body attachment points.
We shall use the default, so select OK from the table.



a- eta/VPG will prompt:
ACCEPT EXTRA NODE COORDINATES?(Y/N)'
b- answer Yes.

8- The suspension model will be displayed based on our defined geometry.

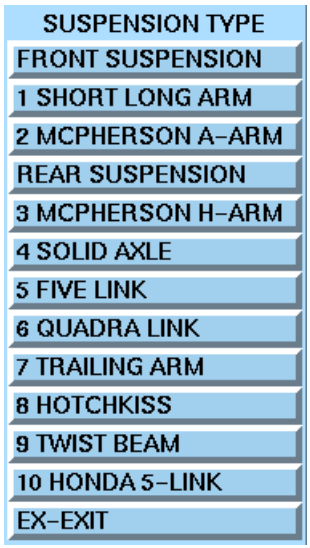


a- the eta/VPG will prompt:
'USE DEFAULT MASS, CG, & INERTIA MOMENT?(Y/N)'
b- answer Yes.

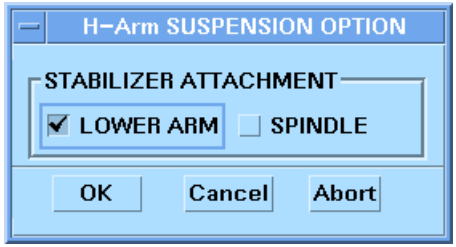
2- Auto-Generating a Rear Suspension Model

1- From the Main menu, select the SUSPENSION menu.

2- Select CREATE. A suspension menu will be displayed from which the user can select different types of front and rear suspension models.

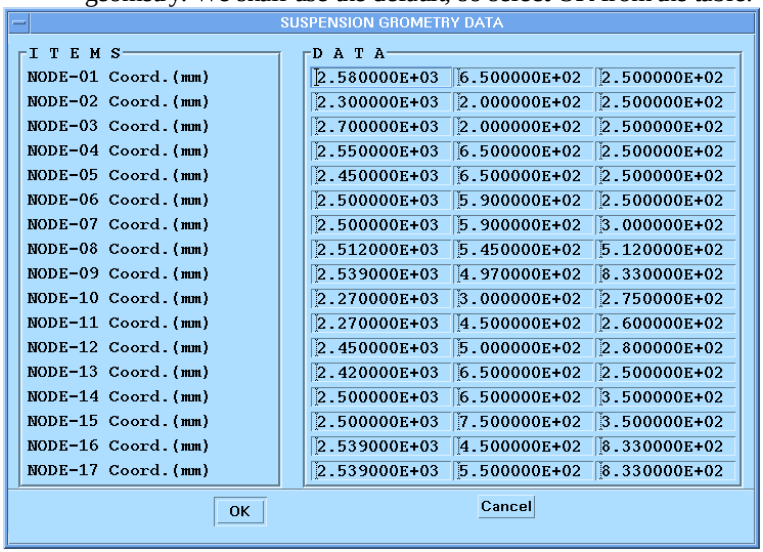


3- For the REAR SUSPENSION, select type:
2- H-ARM



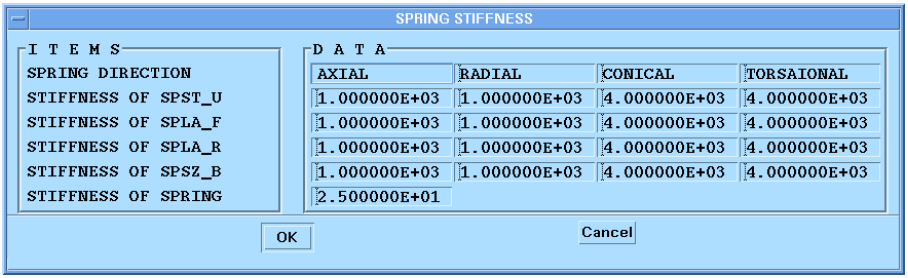
The user can select the Stabilizer attachment point location.

4- The suspension geometry will be displayed along with a table listing the coordinates of nodes that define the geometry. The user can edit this table to define their particular suspension geometry. We shall use the default, so select OK from the table.

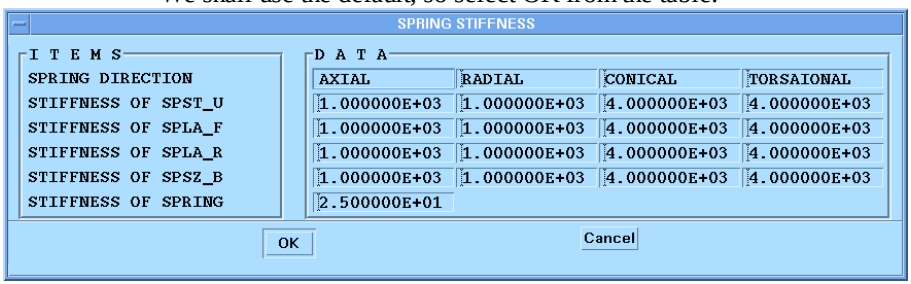


a- eta/VPG will prompt:
'ACCEPT THESE GEOMETRY DATA?(Y/N)'
b- answer Yes.

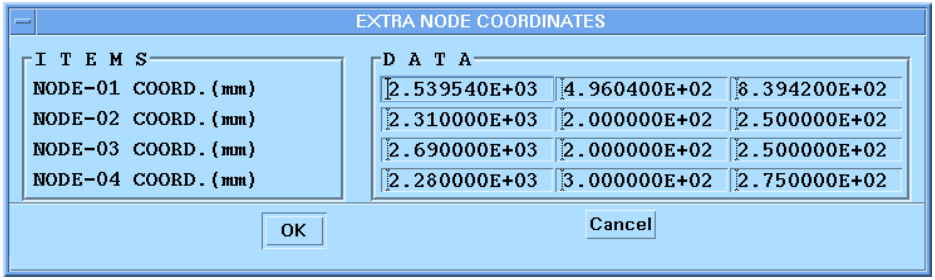
5- A Spring Stiffness menu will be displayed in which the user can specify the spring rates of the suspension. Again, the user can edit this table to define their particular spring rates.
We shall use the default, so select OK from the table.



6- A Damping Coefficient menu will then be displayed in which the user can specify the bushing rates of the suspension. Again, the user can edit this table to define their particular bushing rates.
We shall use the default, so select OK from the table.



7- An Extra Node Coordinate menu will then be displayed in which the user can specify the coordinates of the body attachment points of the suspension. Again, the user can edit this table to define their particular body attachment points.
We shall use the default, so select OK from the table.

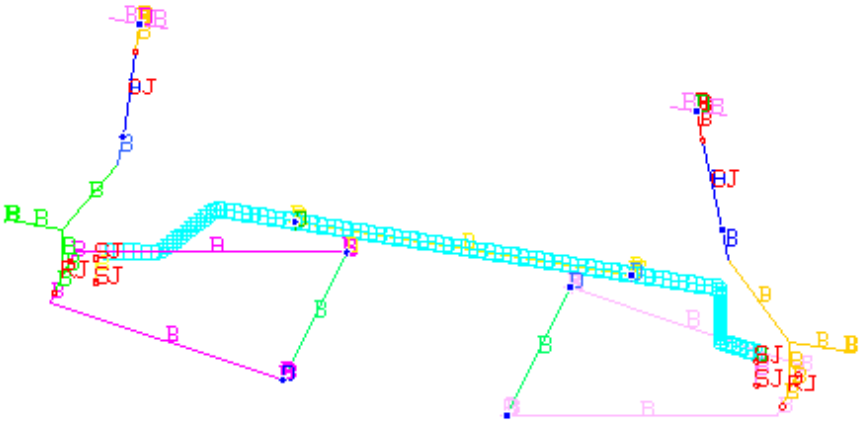


a- eta/VPG will prompt:

'ACCEPT EXTRA NODE COORDINATES?(Y/N)'

b- answer Yes.

8- The suspension model will be displayed based on our defined geometry.



a- then eta/VPG will prompt:

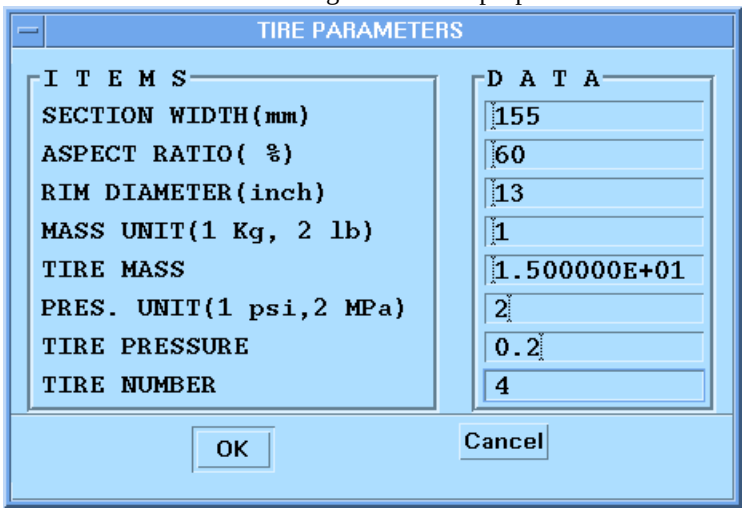
'USE DEFAULT MASS, CG, & INERTIA MOMENT?(Y/N)'

b- answer Yes.

3- Auto-Generating Tire Models

1- From the Main menu, select the TIRE MODEL menu.

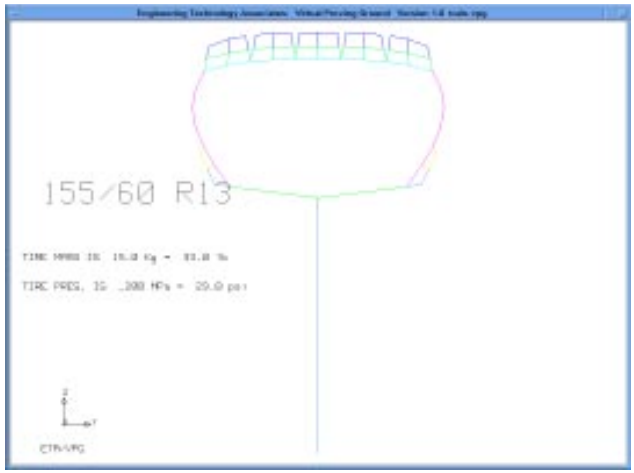
2- Select GENERATE . A tire property menu will be displayed from which the user can change the desired properties.



We shall edit the parameters, except for the pressure value, and the # of tires. Edit the following:

a- PRESSURE UNIT = 2 (MPA)
b- TIRE PRESSURE = 0.2
c- TIRE NUMBER = 4

3- The tire profile will be shown.



a- eta/VPG will prompt:

'DO YOU ACCEPT THIS TIRE PROFILE?(Y/N/ABORT)

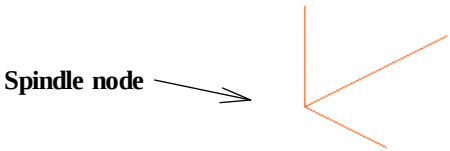
b- answer Yes.

c- eta/VPG will then prompt for tire location:

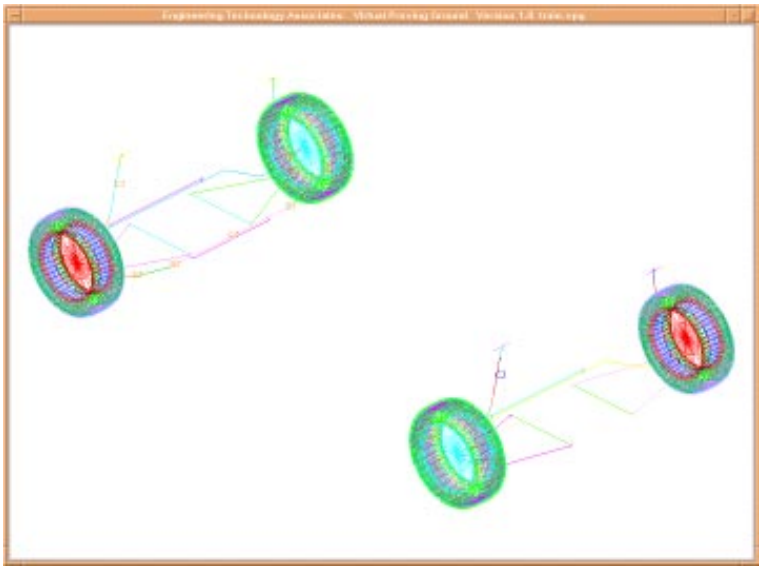
'DEFINE WHEEL CENTER LOCATION FOR TIRE 1'

1- select the NODE option from the Control Keys menu.

2- pick the node at the end of one of the 4 spindles for tire #1.



3- repeat step to define location for the remaining 3 tires(define in a clock-wise or counter-clock-wise direction from tire #1.



- **Attaching Tires to the Suspension**

1- When generating the tires, they will be moved to the ends of each spindle by the user pick.

2- Turn all parts OFF, except the parts containing the wheel hubs(rim) and spindles for each tire.

3- ZOOM into the 1st tire.

4- COPY the 2 coincident nodes that define the middle of each hub and end of each spindle 10 mm outboard(either + or - Y direction depending on which side) of the vehicle.

5- TRANSFORM 1 of the copied, coincident nodes 10 mm to the left or right of the other(so you can see both of them).

6- Go to the ELEMENT options menu and select CREATE/JOINT/REVOLUTE.

7- Use a RELATIVE PENALTY STIFFNESS =1.0

8- Define the joints nodes as follows:

*	*	*	*
1	3	4	2

where: node 1- end of spindle
node 2- free node
node 3- center of hub
node 4- free node

*NOTE: node 2 and 4, it doesn't matter which free node is which.

9- Reverse step 5 to move the free nodes back together(coincident).

10- Repeat the above steps for the 3 remaining tires.

- **Defining Airbags for Tire Pressure**

1- Create a PART SET for each tire(including ALL parts of the tire - the hub(rim)), under PRE-PROCESSOR/SET MENU/PART/CREATE

2- Repeat step 1 for each tire, so you will end up with a total of 4 part sets.

3- Define an airbag for each part set created, under PRE-PROCESSOR/DYNA MISC./AIRBAGS/CREATE/SIMPLE PRESS.-VOLUME.
Edit the following:
a- CARD 1
1- title
2- SID- part set ID

3- SIDTYPE- ID type=1(PART SET)

b- CARD 2

1- CN-Constant Pressure Value (MPA)
or define LCID for time dependent pressure.

4- Repeat step 3 for the remaining 3 part sets.

- **Defining Initial Rotational Velocity**

1- Turn ONLY all 4 tires ON.

2- Create 2 PART_SETs, 1st for the 2 front tires, 2nd for the 2 rear tires.
a- select PRE-PROCESSOR/SET MENU/PART
b- VPG prompts:
'ENTER PART SET NUMBER(DEFAULT) OR E TO EXIT'
'PICK AN ELEMENT OR PART NAME OF A PART'
c- use a MULTI-POINT REGION to select all of the parts of the 2 front tires.
d- select EXIT.

3- Repeat step 2 and define a PART_SET for the 2 rear tires.

*Note: currently LS-DYNA will not run if PART_SETs are used to define Velocity cards, therefore we must define a Velocity card for each part of each tire.

4- Create Local Coordinate Systems for front and rear rotational axes.
a- select UTILITIES/IDENTIFY NODE-POINT to identify the right front tire/spindle attachment node, and the right rear tire/spindle attachment node.
b- note these 2 node numbers and their X, Y, Z coords.

5- Create 2 Velocity cards, 1st for the front tires PSID, 2nd for the rear tires. Select PRE-PROCESSOR/BOUNDARY CONDITIONS/INITIAL CONDITION CARDS/VELOCITY/CREATE/GENERATION.

a- edit the following on CARD 1:

1- CARD DESCRIPTION- user defined.
2- STYP(SETTYPE)= 1(PART SET).
3- ID(ID)- select PART SET ID corresponding to the set number defined in STEP 2.
4- OMEGA(ANGULAR VELOCITY)= road velocity/tire diameter.

b- edit the following on CARD 2:

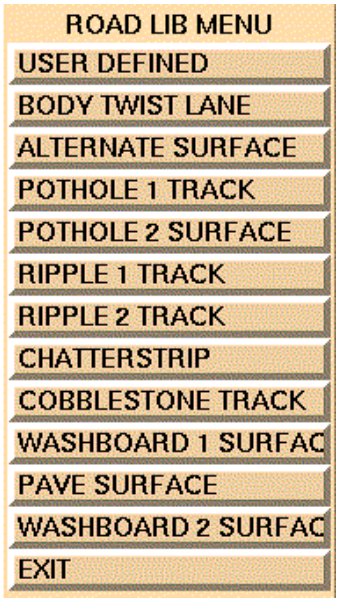
1- Xc- tire center coord for front tire Local Coord. System measured in step 4b.
2- Yc- tire center coord for front tire Local Coord. System measured in step 4b.
3- Zc- tire center coord for front tire Local Coord. System measured in step 4b.
4- Nx=0.0
5- Ny=-1.0(negative or positive depending on velocity sign and Right Hand Rule of LCS).
6- Nz=0.0

6- Repeat step 5 for the rear tire PSID.

4- Auto-Generating a Road Surface

1- Select ROAD from the Main menu.

2- Choose SELECT FROM LIBRARY.



a- select USER DEFINED.

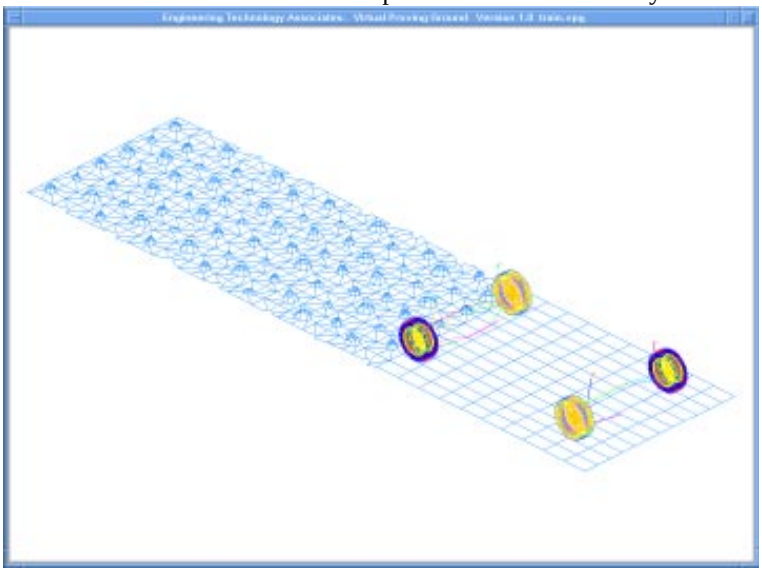
b- VPG prompts:

'ENTER USER DEFINED ROAD FILE OR "STOP" TO EXIT'

1- type in 'road.lib'.

2- select 1-COBBLESTONE.

3- As you will notice, after the road model is read in , the road is located underneath our suspension model. If the road model were not located in the correct position, the MOVE ROAD SURFACE would be used to position the road correctly.



*NOTE: put the model into a SIDE view. Small adjustments may have to be made to the road location by using the PRE-PROCESSOR/NODE/TRANSFORM command to transform the nodes of the road up or down in the global Z direction. The goal is to have 1 to 2 mm gap between the tires and road surface(no penetration). This distance can be checked by using the DISTANCE NODE/PT command in the UTILITIES menu.

- **Defining Contact between Road and Tires**

1- Turn ONLY the outer tread part of each tire and the road surfaces ON.

2- Select PRE-PROCESSOR/CONTACT INTERFACE/CREATE/3-DIMENSIONAL

3- Select 5-NODES_TO_SURFACE interface type.

4- Edit the following:
a- CARD 1
1- CARD DESCRIPTION(eg. tire 1)

b- CARD 2
1- SSID(SLAVE ID) - tire #1 PID
2- MSID(MASTER ID)- road PID
3- SSTYP(SLAVE TYPE)- ID type=3(PART)
4- MSTYP(MASTER TYPE)- ID type=3(PART)

c- CARD 3
1- FS(STATIC FRICTION COEFF.)- 0.7.
2- FD(DYNAMIC FRICTION COEFF.)- 0.7.

5- Use the COPY command to copy the 1st interface card to generate cards for the remaining 3 tires. Edit the CARD DESCRIPTION, CID(contact interface ID), and SSID for each respective tire.

- **Constraining the Road**

1- Turn ONLY the Road Surface ON.

2- Select PRE-PROCESSOR/MATERIAL.

3- Select DEFINE PROPERTIES, and pick the material assigned to the road.

4- The material card appears, edit the following on CARD 2:
a- CON1= 5
b- CON2= 7

*Note: this will constrain the road in the Y, Z and all rotational directions.

- **Assigning Velocity to the Road**

1- Select PRE-PROCESSOR/BOUNDARY CONDITIONS/BOUNDARY CARDS/PRESCRIBED MOTION/CREATE/RIGID BODY.
2- Edit the following to the BOUNDARY_PRESCRIBED_MOTION card:
    a- CARD DESCRIPTION- (e.g. road vel.)
    b- PID(PART ID)- road PID.
    c- VAD(VELOCITY,ACCEL,DISPL)= 0(velocity).
    d- LCID(LOAD CURVE ID):
        1- DEFINE CURVE OPTION/CREATE
        2- CURVE DEFINITION CARD
            a- no editing, select OK.
        3- VPG prompts:
            'ENTER DATA(TIME & VALUE) FOR POINT 1 OR END'
            a- enter: 0, 16000(mm/s).
        4- VPG prompts:
            'ENTER DATA(TIME & VALUE) FOR POINT 2 OR END'
            a- enter: 10, 16000(mm/s).
        *Note: the time value of 10 (secs) is arbitrary, we will not be running the job for that long.
        5- VPG prompts:
            'ENTER DATA(TIME & VALUE) FOR POINT 3 OR END'
            a- type 'end'.
        6- The load curve is displayed along with the curves menu.
        7- EXIT the curves menu.

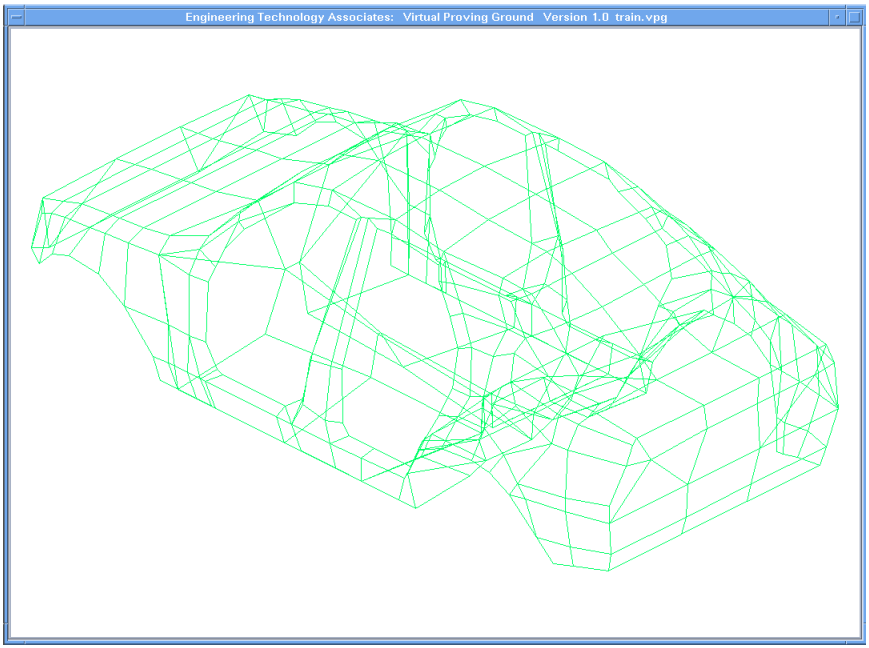
5- Reading the Simplified Body Model

The body to chassis/suspension attachment process depends upon what type of body model the user wishes to use for analysis (deformable or rigid). For this training example, we will use a simple rigid body model to represent the car structure, but instructions will also be given on how to use a deformable body.

1- From the Main Menu, select FILE.
2- Read in the NASTRAN file called 'body.nas'. Again, this is a simplified body model that will be made rigid.

Rigid Body- In this case, we will constrain the rigid body model to the rigid beams that define the body attachment points on the suspension.

Deformable Body- In this case, the specific coordinates for the body attachments points would have be entered when the user defined the Extra Node Coordinates for the front/rear suspension models(procedures 1&2-step 7). This would ensure that the generated suspension would fit the specific body model. The user would create weld spiders between the mounts on the vehicle and the rigid body beams on the suspension.



3- We will assign a rigid material to the body model for this example. Select PRE-PROCESSOR/MATERIAL.
   a- Select CREATE/STRUCTURAL and choose type: 20.1-*MAT_RIGID.
      1- Edit the CARD DESCRIPTION field on CARD to be 'body mat'.
   b- ASSIGN MATERIAL to the body model(the model should turn the color of the property).

5- We will define a new part to contain all of the body attachments on the chassis/suspension. CREATE NEW PART under the PART menu called 'body attach'.
   a- Use the ADD ELEMS TO PART command and move all the beam elements in suspension parts with the name BODY to this new part(body attach).

6- We will now re-assign a rigid material to this new part.

Select any existing RIGID material property card and ASSIGN it to the part 'body attach'.

7- We will constrain the 2 rigid bodies together. Select PRE-PROCESSOR/CONSTRAINT/RIGID BODIES.
   a- Select CREATE and edit the following:
      1- CARD DESCRIPTION - 'body constraint'
      2- PIDM- PID for part containing the body structure.
      3- PIDS- PID for part containing the chassis/ suspension body attachment beams(body attach).

- **Defining Gravity**

1- Select PRE-PROCESSOR/BOUNDARY CONDITIONS/LOAD CURVES/BODY/CREATE/Z-ACCELERATION

2- VPG prompts question:
   'USE PARTS OPTION TO DEFINE CARD?'
   a- select NO, this will apply gravity to the entire model.

3- The BODY LOAD DEFINITION CARD will appear, edit the following:
   a- CARD DESCRIPTION- type in 'gravity'.
   b- LCID(LOAD CURVE ID):
      1- DEFINE CURVE OPTION/CREATE
      2- CURVE DEFINITION CARD
         a- no editing, select OK.
      3- VPG prompts:
         'ENTER DATA(TIME & VALUE) FOR POINT 1 OR END'
         a- enter: 0, 9810.
      4- VPG prompts:
         'ENTER DATA(TIME & VALUE) FOR POINT 2 OR END'

a- enter: 10, 9810.

5- VPG prompts:
'ENTER DATA(TIME & VALUE) FOR POINT 3 OR END'
a- type 'end'.

6- The load curve is displayed along with the curves menu.

7- EXIT the curves menu.

- **Pre-Loading the Suspension**

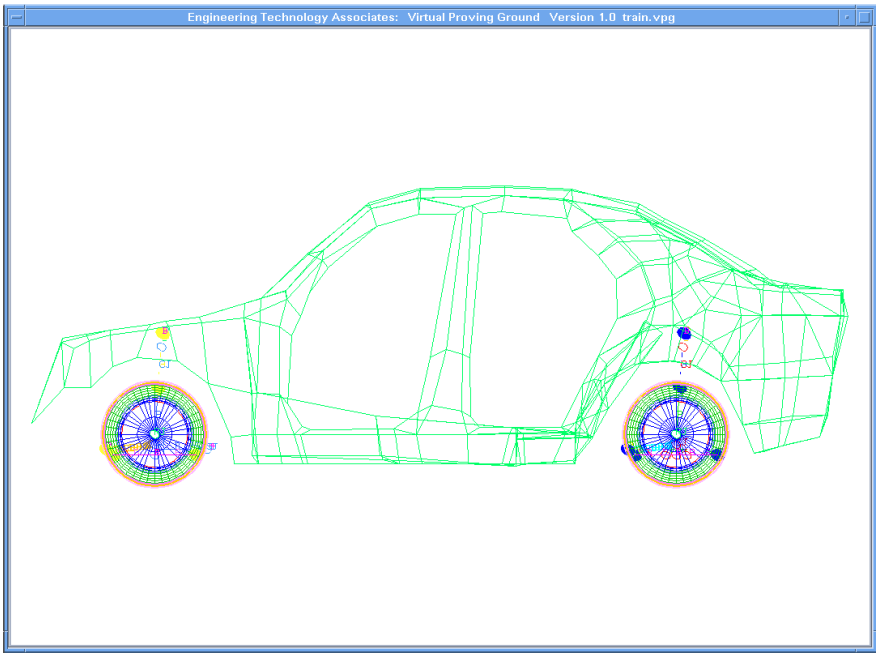
1- Keep ONLY the springs of the suspension On using PART/PART ON/OFF(2 parts named SPRING).

2- Select PRE-PROCESSOR/ELEMENT/MODIFY.

3- Pick an individual spring element.

4- VPG prompts to modify the nodes of the springs:
'SPRING DIRECTION AT EACH END IS IN D.O.F. 0,0'
'PICK NODES/POINTS FOR ELEMENT'
a- select DONE.

5- The ELEMENT DEFINITION card appears, edit the following on CARD 1:
a- INITIAL OFFSET- user defined pre-load amount(-24mm).
*Note: this pre-load amount should be calculated using the body's weight pre spring / the spring rate. The relative weighting of the front and rear suspensions should also be taken into account based on the position of the body's C.G from the front and rear spring attachment points.



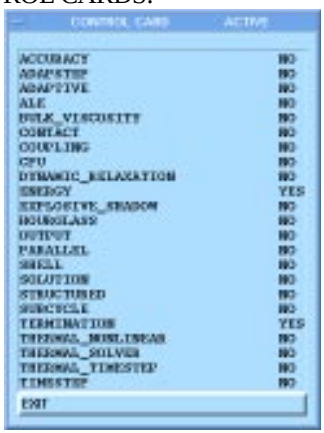
6- Repeat steps 3 thru 5 for the remaining 3 springs.

6- Defining VPG Analysis Control Parameters

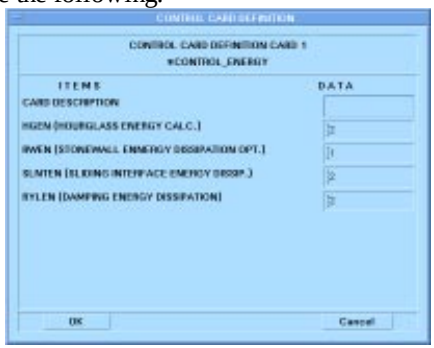
We have completed the model and defined most of the analysis parameters. We will now define the control parameters for the VPG analysis.

1- Return to the Main menu and select ANALYSIS.

2- Select CONTROL CARDS.



a- select ENERGY, then edit the *CONTROL_ENERGY cards' fields to be the following:



b- select TERM, then edit the *CONTROL_TERMINATION card to set the analysis termination time:

3- EXIT back to the ANALYSIS menu, then select DATABASE ASCII. These cards control the output interval for the ascii database files which contain specific analysis result data that can be graphed in the eta/VPG GRAPH PLOT menu.

ABSTAT - AIRBAG STATISTICS NO
AVSFLT - AVS FORMAT DB NO
BNDOUT - BOUNDARY CONDITION NO
DEFGEO - DEFORMED GEOMETRY NO
DEFORC - DISCRETE ELEMENT YES
ELOUT - ELEMENT DATA NO
GCEOUT - GEOM. CONTACT ENT. NO
GLSTAT - GLOBAL DATA YES
JNTFORC- JOINT FORCE DATA YES
MATSUM - MATERIAL ENERGIES NO
MOVIE - MOVIE FORMAT DB NO
MPGS - MPGS FORMAT DB NO
NCFORC - NODAL INTERFACE NO
NODFOR - NODAL FORCE GROUPS NO
NODOUT - NODAL TIME HISTORY YES
RBDOUT - RIGID BODY DATA YES
RCFORC - RESULT INTERFACE YES
RWFORC - RIGID WALL FORCES NO
SBTOUT - SEAT BELT DATA NO
SECFORC- SECTION FORCES NO
SLEOUT - SLIDING INTERFACE NO
SPCFORC- SPC REACTION FORCE NO
SSSTAT - SUBSYSTEM DATA NO
SWFORC - SPOTWELD FORCES NO
EXIT

a- select DEFORC and edit the output interval time to be:

b- select GLSTAT and edit the output interval time to be = 5.0e-3 (sec).

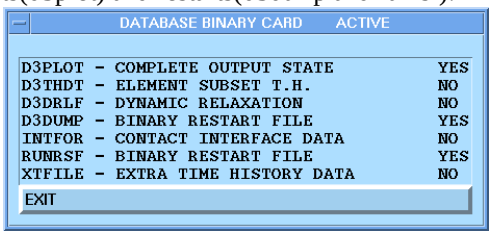
c- select JNTFORC and edit the output interval time to be = 5.0e-3 (sec).

d- select NODOUT and edit the output interval time to be = 5.0e-3 (sec).

e- select RBDOUT and edit the output interval time to be = 8.0e-3 (sec).

f- select RCFORC and edit the output interval time to be = 5.0e-3 (sec).

4- EXIT back to the ANALYSIS menu, then select DATABASE BINARY. These cards control the output interval of the binary database files for results(d3plot) and restarts(d3dump and runrsf).


DATABASE BINARY CARD ACTIVE
D3PLOT - COMPLETE OUTPUT STATE YES
D3THDT - ELEMENT SUBSET T.H. NO
D3DRLF - DYNAMIC RELAXATION NO
D3DUMP - BINARY RESTART FILE YES
INTFOR - CONTACT INTERFACE DATA NO
RUNRSF - BINARY RESTART FILE YES
XTFILE - EXTRA TIME HISTORY DATA NO
EXIT

a- select D3PLOT and edit the output interval time to be:

b- select D3DUMP and edit the output frequency to be = 9.99E+08.

Users should be aware that these files can be fairly large, and since consecutive files are written based on the frequency(i.e. d3dump, d3dump01, d3dump02), users have to be careful not to fill up their hard disk.

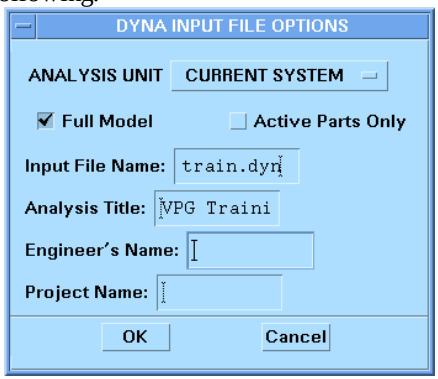
c- select RUNRSF and edit the output frequency to be = 5.0E+03.

Unlike the d3dump file which is written consecutively, the runrsf file is overwritten based on the frequency(only 1 runrsf will exist, although it will increase in size each time). Therefore, using this file is safer then using the d3dump file is hard disk space is a consideration.

7- Analysis Submission

Now that we have completed defining our analysis control cards, we can submit the analysis.

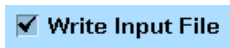
1- From the ANALYSIS menu, select DYNA INPUT FILE OPTIONS, and edit the following:



a- Input File Name: train.dyn
b- Analysis Title: VPG Training
c- Engineers Name: (your name here)

2- The user can now select whether to submit the analysis directly from the window, or write out the input file without submission. LS/DYNA should be installed and setup on the workstation to submit analysis directly from the eta/VPG window. We will write out the input file and take a look at how it is setup, and because of time requirements we will not run the analysis. We will post-process results that have been already run.

a- click on WRITE INPUT FILE, then click OK. The LS/DYNA input file will be written out with the name 'train.dyn'. The user should 'vi' this file to see how it has been setup.



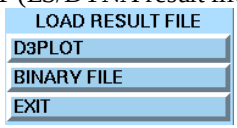
Note: The user should reference the eta/VPG User Processor Manual for information on how to submit the analysis from outside eta/VPG, and how to use the control (Sense) switches when LS/DYNA is running. The user should also reference how to Restart files(d3dump and runrsf), and reference the terminology of other various LS/DYNA files.

8- Displaying Results

We will now post-process the results from our analysis. RESTART the eta/VPG session, and CREATE A NEW FILE called 'pp.vpg'- pp for post-processing.

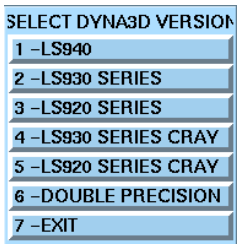
1- From the Main Menu, select POST-PROCESSING.

a- select D3PLOT (LS/DYNA result file).

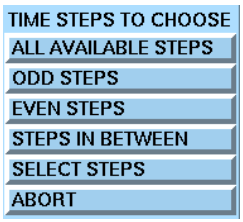


b- select the file 'd3plot' from the File Menu.

c- select LS/DYNA version 940



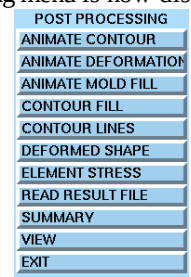
d- select ALL AVAILABLE STEPS



e- select ALL COMPONENTS

The results from each of the 10 steps will then be read into eta/VPG. A binary result file will be created at this time(named ‘d3plot.pp’). Since the results are not saved to the eta/VPG database, this binary file should be re-read into the post-processing menu when the user wishes to view the results again. This file is read much faster than the d3plot files.

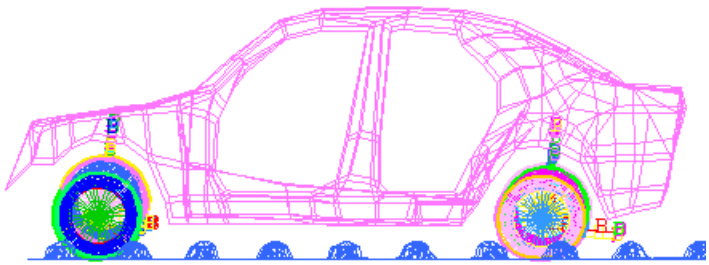
2- The Post-Processing menu is now displayed.



3- Put the model into an ISO-metric view, and select ANIMATE DEFORMATION.

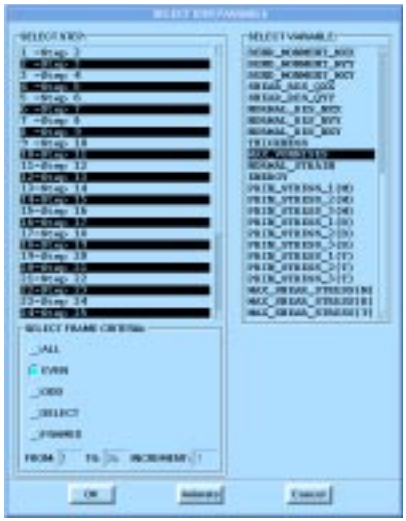
4- Select EVEN steps from the menu, and then ANIMATE.

5- Select SIDE view to view the suspension reaction.

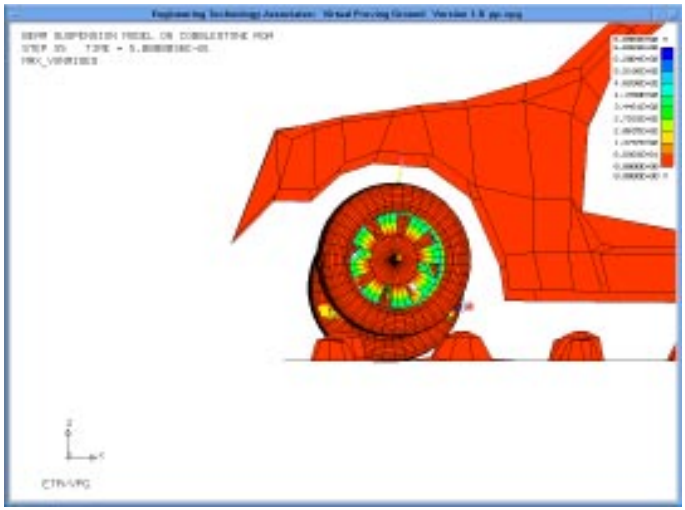


6- EXIT out of the animation menu, then select ANIMATE CONTOUR.

7- Select EVEN steps and MAX_VONMISES contour variable. Then pick ANIMATE.



a- ZOOM into the front tires to see the stress distribution in the wheel. (we will not see any stress contours in the body model, again, this is because the body is rigid in this analysis).



During the animation, the user can select INDIVIDUAL FRAME to scroll thru each result frame, increase/decrease the animation speed, TURN PARTS ON/OFF, and SAVE MPEG files which can be played on most PC's and workstations without having the eta/VPG program.

8- Select DISPLAY OPTION:

Turn the ELEMENT OUTLINE off, change the CONTOUR BAR ORIENTATION to HORIZONTAL, change the CONTOUR LEVEL to 25, and change the UPPER RANGE to 400 MPA, then re-ANIMATE.

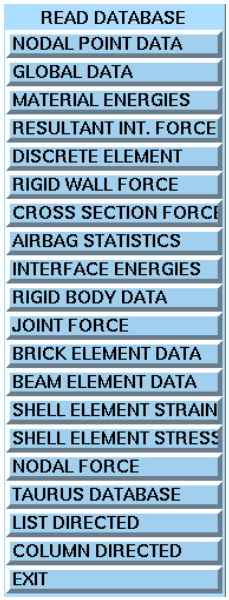
DISPLAY OPTIONS
ELEMENT OUTLINE
SHADING
DEPTH CUEING
SHOW CONTOUR MAX./MIN. MARKS
PLOT TITLE
REVERSE CONTOUR COLOR
CONTOUR COLOR:
CHANGE...
RESET
CONTOUR BAR ORIENTATION:
VERTICAL
HORIZONTAL
CONTOUR LEVEL:
CONTOUR RANGE
UPPER AND LOWER RANGE:
.00
LOWER
400.00
UPPER
DEFORMATION SCALE FACTOR:
OK
Animate
Cancel

9- Graph Plotting

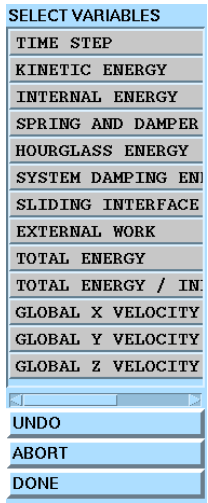
1- We will not plot some LS/DYNA ASCII database results graphically. EXIT out of the Post-Processing menu.

2- Select GRAPH from the Main Menu, then select CREATE NEW FILE from the FILE MENU.

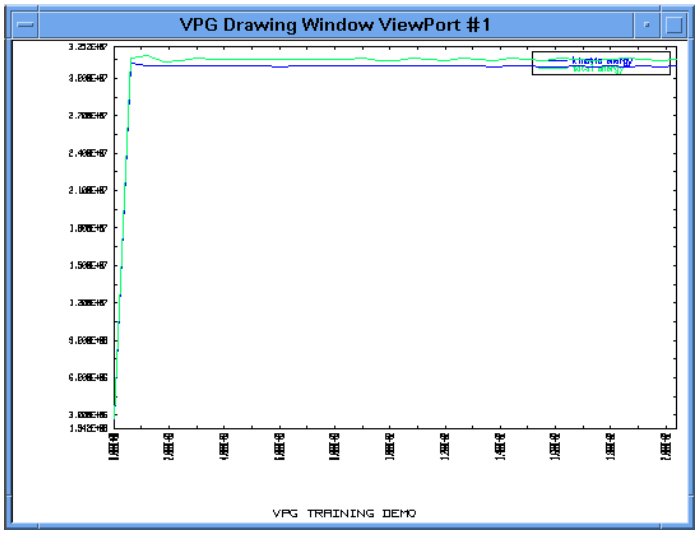
3- Select GLOBAL DATA, and choose the filename ‘glstat’ from the FILE MENU.



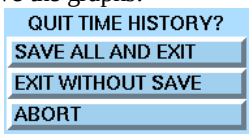
4- Select KINETIC, and TOTAL ENERGY from the variable list.



5- The graph of the energies will be displayed. The TIME HISTORY PLOTS menu can then be used to manipulate the graph(s) and also read more graphs into the database (a total of 16 graphs can be read into an individual database).



6- Upon EXITING the TIME HISTORY PLOTS menu, the user will be prompted to save the graphs.



a- if the user selects to save the graphs, they will be prompted:
'PLEASE INPUT BINARY FILE NAME <fembth.gr>'

b- the graph info. will be saved to this file, and like the d3plot.pp file, the graph file can be re-read at a later time without having to re-read the database file(s).

Conclusion

This concludes the eta/VPG training manual. The user should now have a basic understanding of how to create an eta/VPG model, setup the analysis and post-process the results. Reference the Applications Manual for specific information about analyses, and the User's Manual for more detailed description of commands.